CAD in Europe (1)

a review of schematic drawing and PCB layout programs

By Guy Raedersdorf

If there's any field where developments come at a lightning pace, it's without doubt the realm of computer programs, especially those for the PC. The number of programs that are intended to make life easier for the electronic hobbyist just keeps on growing. We thought it would be a good idea to review several European programs for generating schematic drawings and PCB layouts.

First, a preliminary remark: in our opinion, the subject of simulation warrants an article of its own, which will perhaps appear within a few months. This is why we limit our attention in this article to programs for generating schematic diagrams and PCB layouts.

Our adventure started with sending e-mail messages to all of the publishers of these types of programs that we were aware of and that we considered including in our review. We expected to be able to relax for a few days after that, but positive responses came streaming in from the next day onwards.

It is not our intention to make any sort of comparative study followed by a classification and the awarding of points. Instead, we want to present the various products to you and describe the specific features that attracted our attention in the short time that we used them. In the case of products that are distributed via the Internet, we also want to tell you where to find them.

It is humanly impossible to summarise all the features of a program in a few column-inches of text. Consequently, we must ask you to forgive us for a certain amount of 'superficiality'.

In the light of the number of responses that we received, we feel compelled to split this article into two parts. The first part, which is the article you are now reading, is dedicated to three programs that we received in complete versions, to wit:

- Multisim + Ultiboard
- Eagle Layout Editor
- Electronic Design Studio

The second article of the series deals with programs that we received as demo, evaluation or limited versions that we were able to access via the Internet.

Multisim and Ultiboard

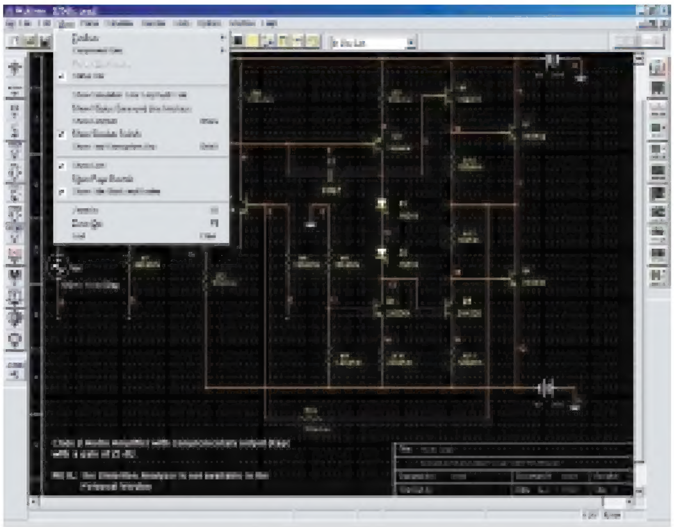
Multisim

- Source: Electronics Workbench
- Version tested: Multisim 2001
- Medium: CD-ROM + serial number
- Language: English
- Installation: two steps
- De-installation: clean and easy
- Documentation: one thick, carefully prepared manual (in English)
- Price: from 399 ex VAT (personal version) to 4995 ex VAT for the Multisim Power Pro version with Verilog (Ultiboard is available as an option for both versions).

Ultiboard

- Source: Electronics Workbench
- Version tested: 5.63
- Medium: CD-ROM + serial number + dongle
- Language: English
- Installation: simple
- De-installation: quick and clean
- Documentation: 1 extremely thick, well-prepared manual

Internet addresses:
www.electronicsworkbench.com
www.interactiv.com



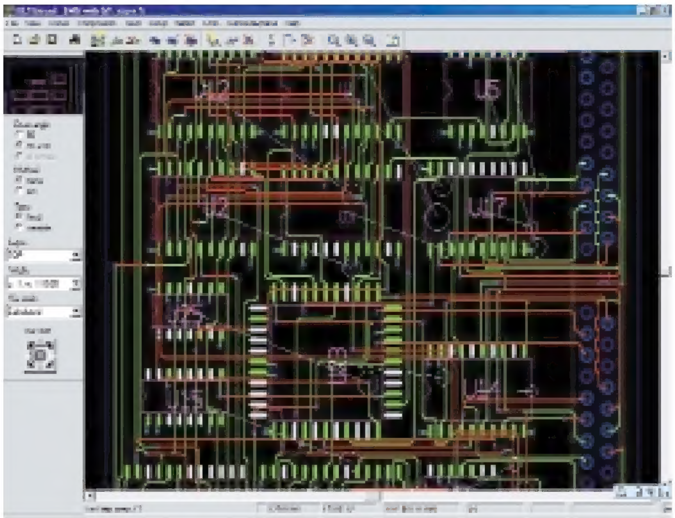
The first programs that we received were the Multisim and Ultiboard duo from Electronics Workbench.

Multisim is a complete development tool that also includes elements falling within the scope of this article, namely a schematic drawing package with a very extensive component library. These programs also include other elements that fall outside our current scope, namely SPICE (a complete analogue and digital simulation program with a schematic drawing package and a VHDL/Verilog simulation package, FPGA/CPLD analysis and RF capability), post-processing options and problem-free coupling to layout development programs such as Ultiboard. As can be seen from the screen dump, there is a familiar schematic drawing environment with menus, Multisim toolbars and a component bar on the left-hand side. Placing components is simple — they are automatically aligned to the grid — and making connections is child's play. Actually, generating schematic drawings for electronics projects is the axis about which everything else revolves.

The (relative) complexity of Multisim is related to the fact that it is in principle a simulation program that has been equipped with schematic drawing facilities and many other tools that are more specifically intended to be used for simulation.

You should not forget that Multisim, like many of the competing programs, allows you to modify existing components and even to create new components. The standard library contains 16,000 components. It would go too far to describe them in detail.

Next, let's look at **Ultiboard**, which is the PCB layout program. This program works with a dongle.



The installation of Ultiboard takes almost no time at all. It is not even necessary to restart the computer.

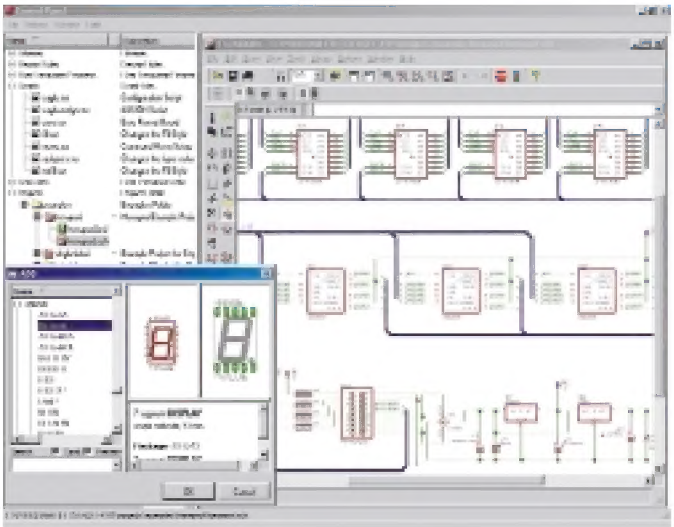
Ultiboard is a very complete layout program. There is a link to the schematic drawing program, which normally includes a simulation model and footprint for each component. This considerably simplifies the use of this powerful pair of programs.

In conclusion, if all you want to do is to generate schematic drawings and PCB layouts, there are simpler solutions, but if you also want to check your schematics using simulations, you will need the Multisim module of the Multisim + Ultiboard duo.

Eagle Layout Editor

- Source: CadSoft (Germany)
- Version tested: 4.01
- Medium: CD-ROM + key disk
- Language: English
- Installation: simple
- De-installation: no problems
- Documentation: two manuals (in English) totalling 250 pages.
- Price: from 96 DM (incl. VAT) for the Light version to 2723 DM (incl. VAT) for the Professional version.

Internet address:
www.cadsoftusa.com



In case you are wondering where the name 'Eagle' comes from, it is an acronym for 'Easily Applicable Graphical Layout Editor' — that sounds promising!

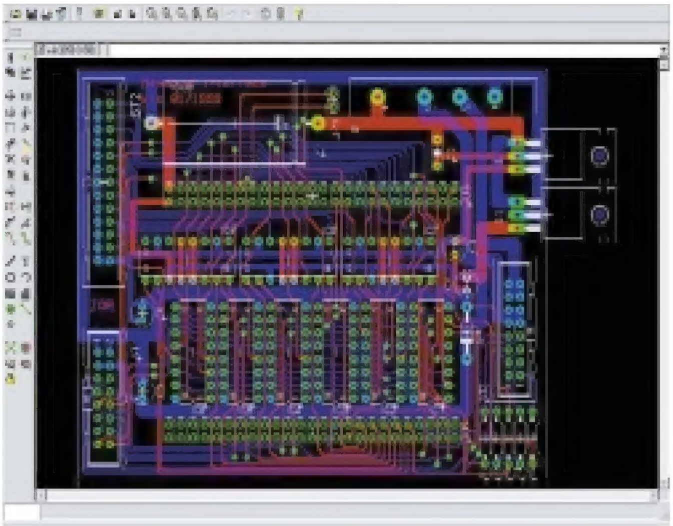
A few special features of Eagle:

Updated component management with a browser for viewing the various components.

A choice of different footprints, depending on the function and technology of the component.

Very up-to-date documentation, in the form of both printed material and PDF files.

A scripting language is included, to allow a number of routine procedures to be recorded.



The menu for the various functions can be seen as a tree structure on the left-hand side of the screen dump.

A stroll through the libraries is impressive. The Atmel library, for instance, has more than 50 entries for footprints and components. When you select a component, you can thus immediately specify the footprint belonging to that component.

You should not let anything hold you back from getting your hands on this program.

The tools for the schematic drawing and layout programs are extremely complete.

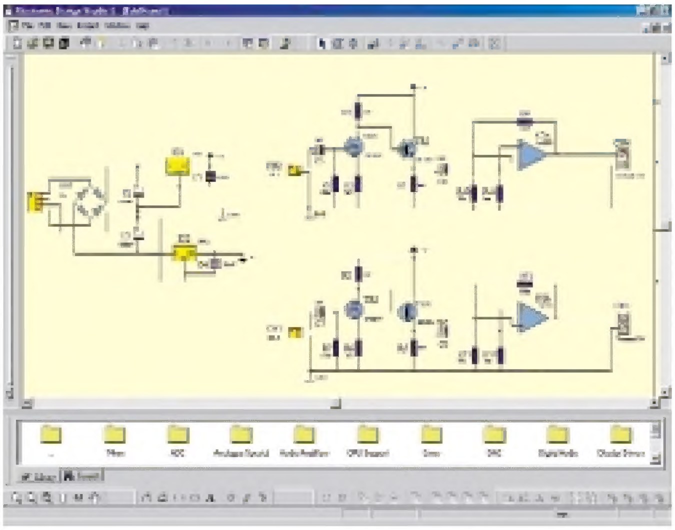
The final thing that we should mention is that **Eagle 4.0 Light** is available as freeware for non-commercial use. The program can be used with the accompanying freeware key for evaluation or personal use.

We can conclude that **Eagle** is very user-friendly; with this complete program, you will be able to generate very nice schematic drawings and layouts.

Electronic Design Studio

- Version tested: EDS2 2.0D Advance
- Medium: CD+ key disk
- Language: English
- Installation: easy
- De-installation: faultless, but don't forget to delete the key!
- Documentation: two manuals (in English) totalling 130 pages
- Price: from £199 ex VAT for the Standard version to £349 ex VAT for the Advance version.

Internet address:
www.quikroute.co.uk



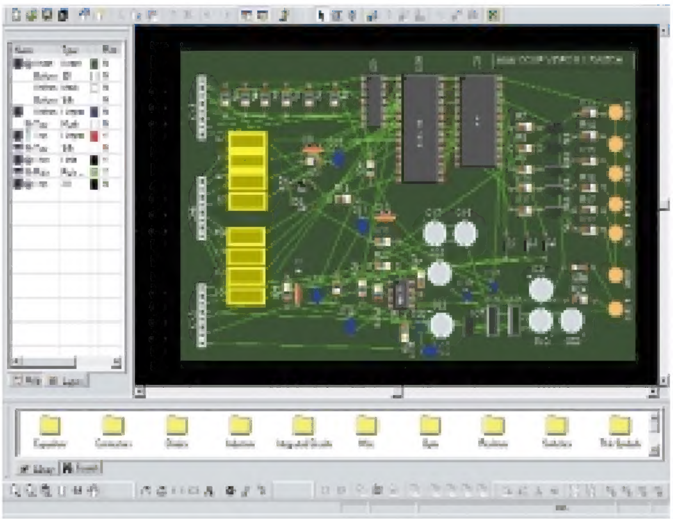
The product range of Quickroute Systems Ltd consists of four versions of their schematic drawing and layout program: **EDS2 Lite, EDS2, EDS2 Advance and EDS Ultra.**

The differences relate to the features of the layout module. The three simplest versions can handle 32 layers, while there is no limit on the number of layers in the Ultra version. The Advance and Ultra versions have a 'rip-up and retry' function, a router that can handle multiple tracks with different widths at the same time and some other handy features, such as the creation of copper planes. The Lite version does not have the routing and simulation features. The resolution of the first three versions is 1 μm (1 micron), while the Ultra version has a resolution of 1 nm (0.001 μm).

As with many of the other programs presented here, Electronic Design Studio can lay claim to being a modular, integrated electronics CAD system.

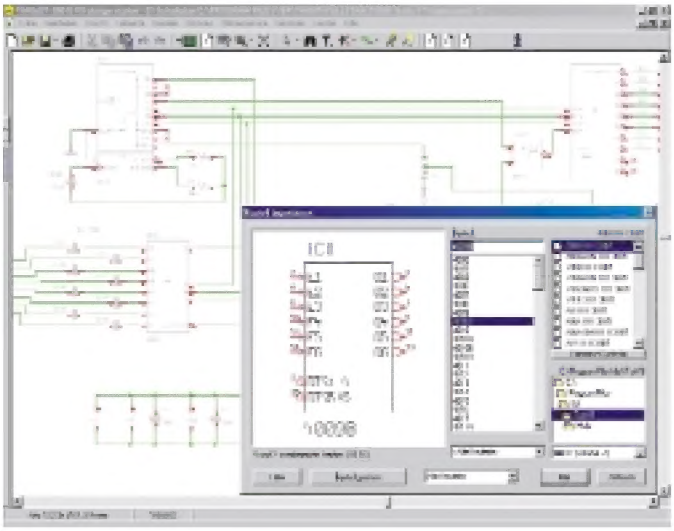
In the screen dumps, you can see that this is a professional-level schematic drawing and layout program. Just have a look at the 3-D form of the components on the printed circuit board.... The CAD/CAM printout driver also supports TrueType fonts. The schematic drawing printouts are tidy and certainly do justice to the new millennium.

With the exception of the Lite version, it is possible to work



with several sheets, each of which can be up to 2 m by 2 m (!). Components can be easily placed on the document, and numbering is automatic. EDS also automatically generates a netlist.

The three most powerful versions have a Viper autorouter. EDS automatically generates the printed circuit board layout from one or more schematic drawings or a netlist. The components are placed on the circuit board, and the interconnections are visible as a 'rat's nest'. The user can reposition the components and do manual routing, automatic routing or both together.



Although it is outside the scope of this article, we would like to mention that all versions except the Lite version have a simulation function. The package contains a coupon that can be redeemed for a Eurocard format film made from a Quickroute design.

EDS2 on the Internet: the version that we downloaded from the Quickroute requires a serial number, which you can obtain via e-mail or fax.

Important: before reformatting your hard disk, don't forget to move the key back to the diskette. The manual has an excellent description of how to do this. Make sure that the diskette is not write protected, since otherwise the program will refuse to start and an error message will appear.

With the description of the last of these four programs, we have nearly arrived at the end of the first article.

In order to avoid any misunderstanding, we should point out that the schematic drawings and circuit board layouts that can be seen in the screen dumps were not generated by the author. They are examples taken from the samples and tutorials delivered with the various programs.

Conclusion

It has not escaped our attention that over the years, certain collaborative arrangements have arisen between previously independent companies. Some examples are Ultimate (known for its Ultiboard and Ulticap programs), which has joined with Electronics Workbench, and OrCAD, which has merged with Cadence. There is no doubt that we can expect to see other 'marriages' in the future.

It is also entirely possible that even as you read this article, a new version of one of these programs will have already appeared, since everything moves super-fast in the ever-changing CAD world.

In conclusion, we would like to make one general remark. We see ourselves surrounded by programs that are steadily becoming more complex. This means that regardless of which program we are dealing with, we must take the time to read the documentation and to practice with simple circuits before tacking more complicated tasks. You certainly should not regard time spent in looking at examples (which are usually contained in a 'Tutorial' subdirectory) as being wasted.

(000148-1)

Next month's instalment will introduce the rest of the schematic drawing and printed circuit board layout programs, which we received in the form of demo, time-limited or evaluation versions.

In random order, these programs are:

- SpicycleTM
- Easy-PC for Windows
- OrCAD Capture & OrCAD Layout
- Layo PCB
- Target 2001! V8.2F
- Proteus VSM
- Protel 99SE
- p-cad® 2000
- Ranger XL
- CIA04
- CAD PACK
- WINSCHEM & WINTYPON
- Windraft
- Winboard
- VUTRAX
- Edwin

A whole collection, as you can see!